

International Conference on Manufacturing Engineering and Materials, ICMEM 2016,  
6-10 June 2016, Nový Smokovec, Slovakia

## Influence of the individual layers of laminate on the final static response

Michal Halapi, Zuzana Murčinková

*"Department of Technical Systems Design, Faculty of Manufacturing Technologies with seat in Prešov, Technical University of Košice,"*

### Abstract

The thesis based on a model example, describes in details the formulation of composite materials in simulation software. Furthermore, it points out the influence of the material properties orientation in composite materials by means of a comparison between two geometrically identical models with different material properties orientations. Changes in stress of layers were achieved through changes in orientation. The results of the achieved maximum stress levels are systematically processed in the table, in which the influence of orientation across the individual layers may be clearly observed.

© 2016 Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

Peer-review under responsibility of the organizing committee of ICMEM 2016

**Keywords:** "FEM, composite, numerical simulation, orientation of material properties, lamina"

### 1. Introduction

Composites are modern but not universal materials. A composite material is a material made two or more constituent materials with significantly different physical or chemical properties that, when combined, produce a material with characteristics different from the individual components. They are always designed for a specific use with the aim of their maximum matter usage. Composites are considered to be versatile primarily thanks to their high measurable solidity and firmness, favorable fatigue limit and the fact that they are rather frost-proof. Composites represent a qualitative change in solving the contradictory issues between required characteristics and capabilities of homogenous materials. It is often needed to take into account requirements which are truly contradictory and non-achievable by the properties of one homogenous material. The choice of possible combination of the materials is, therefore, a compromise which arises from fulfilling the primary requirement on the final properties of components. Composites are direction-oriented materials, what mean that the material properties orientation shall influence them. [2,3,4,6]

### Nomenclature

$^{\circ}$	Degree of angle
$E_1, E_2, E_3$	Young Module
FEM	Finite element method
$F_1, F_2$	Force
$G_{12}, G_{13}, G_{23}$	Shear module
mm	Millimeter
MPa	Mega pascal
$\nu_{12}$	Poisson number
N	Newton

\* Corresponding author. Tel.: +421-556-026-315

E-mail address: [michal.halapi@tuke.sk](mailto:michal.halapi@tuke.sk)

## 2. Simulation of fiber composite materials

A computer simulation using FEM is not a simple procedure at homogenous materials, at composites the calculations are even more demanding. The formulation and evaluation of a composite material model from final elements at the same geometry is more time-consuming than of a model of isotropic material. When using orthotropic material, it is needed to define the material orientation. Layered elements, shell elements and the number of integration points at the thickness of the shell, they all result in a greater amount of data in comparison with a non – layered material. This fact is associated with a more demanding results evaluation. There are several types of software that are successful in solving these issues; one of them is Abaqus/CAE, which we used. Abaqus/CAE offers two types of shells for modeling the composites:

- Conventional shell – a geometry modeled on reference area, which does not have to be identical with the centring area of the shell. This one offers the option of specification for the lower, middle and upper surface.
- Continuum shell – this one can work with continuum geometry, where the thickness will be determined by the geometry. When defining the composition of the laminate only relative thickness of laminas are entered to the total thickness. [1,10,11,]

## 3. Computing model formulation protocol

When evaluating the influence of single layers of laminate on the final properties of a composite material the calculations were done for a simple-shaped model. The exact procedure of the computing model formulation can be found in the following:

- The model geometry designing – geometry that might be more specifically defined as 3D, compliant and shell body was formed in the module Part. The parameters of the used model may be seen in Fig. 1, the protrusion of the shown design being 40 mm.

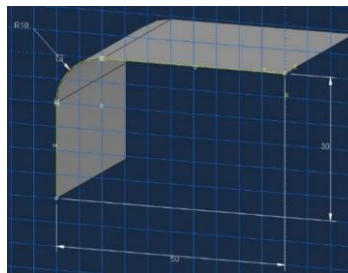


Fig. 1 Geometry of numerical model

- Coordinates system definition – as the materials are direction-oriented, it was needed to define the coordinates system. As the coordinates system according to which the fibers were supposed to get oriented is identical to the global coordinates system, it was not necessary to create a new one. The software managed to get the material properties oriented correctly thanks to assigning the coordinates system orientation to the model. The long-fibered composites, which show orthotropic properties, have three mutually perpendicular plains of the material properties symmetry. The plains are determined by pairs of orthogonal axes labeled as 1, 2, and 3, so called material axes. The axis 1 – alongside the fibers, the axes 2, 3 – perpendicular to the fibers. The highest solidity may be, therefore, seen in the direction of the axis 1.

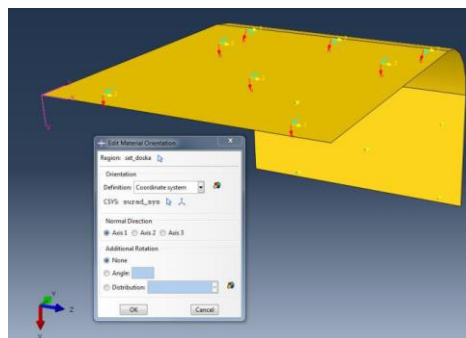


Fig. 2 Definition of coordinate system

Coordinates system shown in Fig. 2, on the basis of which the individual layers are oriented in dependence on the angles set in the Fig. 3 and Table 1.

- Production of materials – it is essential to correctly set the material constants of the used materials. The software employs the characteristics by means of: Young module ( $E_1$ ,  $E_2$ ,  $E_3$ ), Poisson number ( $\nu_{12}$ ) and shear module ( $G_{12}$ ,  $G_{13}$ ,  $G_{23}$ ) in the individual axes and plains. Constants needed for defining the material were entered to the software according to the material definition. The values of those constants may be either filled in based on the data sheets, or defined by software calculations. It is a fact that when the geometry of a model is given in mm, the units of these constants are given in MPa and the results will also be shown in MPa.

Table 1 Material properties

Material	Core	Lamina
Density	8E-011	1,5E-009
E1	10	35000
E2	10	7500
Nu12	0,3	0,3
G12	1	3600
G13	30	3000
G23	30	3000

- Assigning the materials to the geometry of the model – this step includes determining 11 layers of the laminate, their thickness, material used, orientation towards the coordinates system. The defining position of the layer towards the core, which is in this case a base onto which all the other layers were placed, is also dependent on the geometry of the model. The individual layers with their positions (over or under) towards the reference area Fig. 1, materials, thickness of the individual layers and their orientation is assigned to the reference area (set\_doska) according to the selected coordinates system.

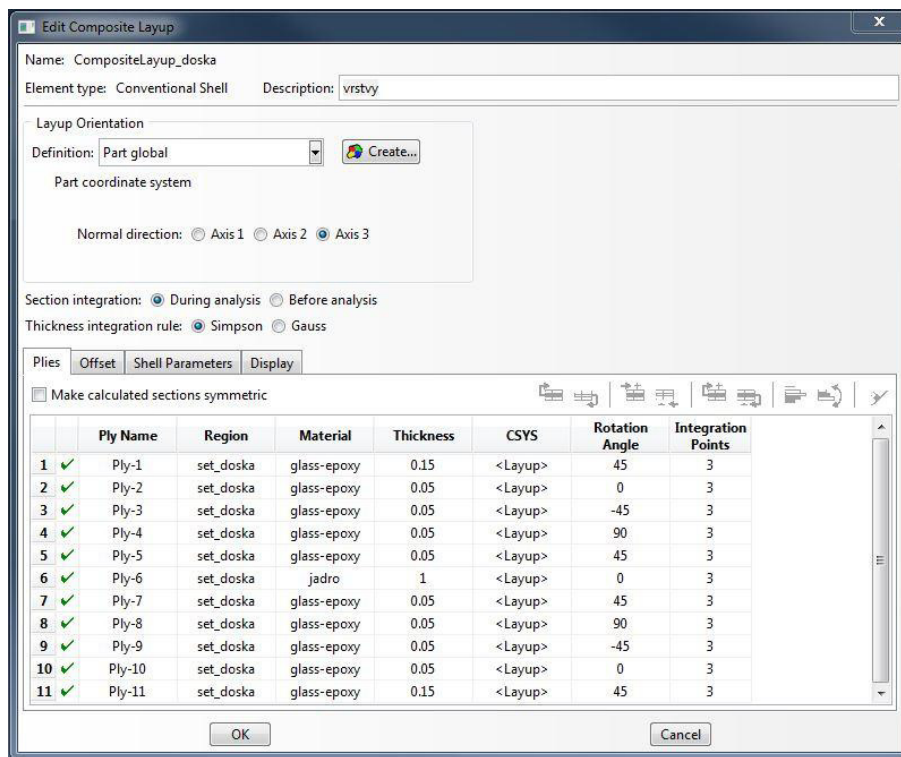


Fig. 3 Arrangement of individual layers in the model

- The model composed in this way is the given computing conditions in the Step module. The task was solved as static and non-linear with the total length of the step 1, size of the starting increment 0,05 and maximum length 0,1. The desired output of this analysis was to obtain the stress amount of Mises.
- The module Interaction was used to determine the bounds. The bond Coupling was assigned to the smaller rectangle area, as may be seen in the Fig. 4.
- Boundary conditions were set in the module Load, the module saved by the constraint in all the six directions was burdened by the forces pair  $F_1 = 50$  N in the direction of axis  $x$  and  $F_2 = 80$  N in the direction of axis  $y$  Fig. 4, which acted in the mutually perpendicular direction. Global coordinates system identical to the coordinates system for material properties orientation was used in this case.

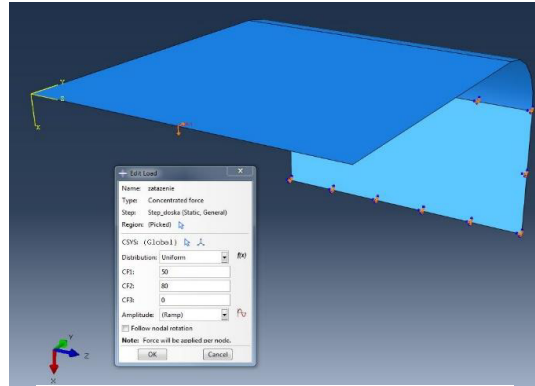


Fig. 4 Load of clamped model

- Mesh model was created in the module Mesh by means of shell elements S8R – quadratic elements with reduced integration. The global element size was set to 0,3 mm.

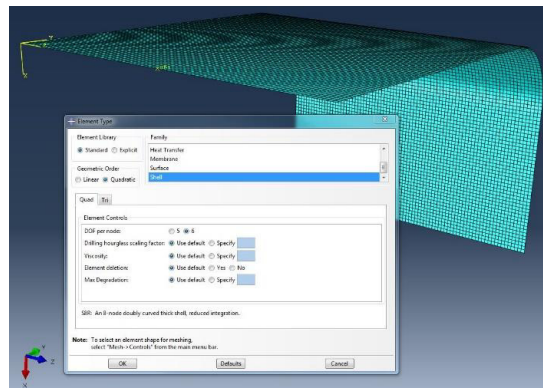


Fig. 5 Mesh model with use S8R

#### 4. The influence of the configuration on the final stress

This chapter includes the results of the influence of the layers orientation on the final stress. The other model was designed in the same manner, only the arrangement of the layers was different. The extent of stress for the individual layers in dependence on the angle of rotation towards the selected coordinates system may be seen in the Table 2. All the other layers parameters were kept so as the influence of the orientation change may be shown.

Table 2 Achieved results

Ply	Max. Mises [MPa]	Angle 1 [°]	Max. Mises [MPa]	Angle 2 [°]	Thickness [mm]	Material
Ply 1	310	45	643,9	-60	0,15	Glass – epoxy
Ply 2	658,8	0	1390	-30	0,05	Glass – epoxy
Ply 3	338,8	-45	733,6	0	0,05	Glass – epoxy
Ply 4	528,4	90	1000,2	60	0,05	Glass – epoxy
Ply 5	212,1	45	442,1	30	0,05	Glass – epoxy
Ply 6	0,046	0	0,087	0	1	Core
Ply 7	199,3	45	405,7	30	0,05	Glass – epoxy
Ply 8	398,7	90	781,3	60	0,05	Glass – epoxy
Ply 9	327,4	-45	694,7	0	0,05	Glass – epoxy
Ply 10	599	0	1256	-30	0,05	Glass – epoxy
Ply 11	297,2	45	607,5	-60	0,15	Glass – epoxy

On the Table 2 and Fig. 6 are results of tension in each plies for both calculation models. The maximum tension at the first calculation model have been at ply 2 with angle of rotation 0 degrees, the second computation model was achieved maximum tension also on ply 2 with angle of rotation -30 degrees. Different of the tension is considerable.

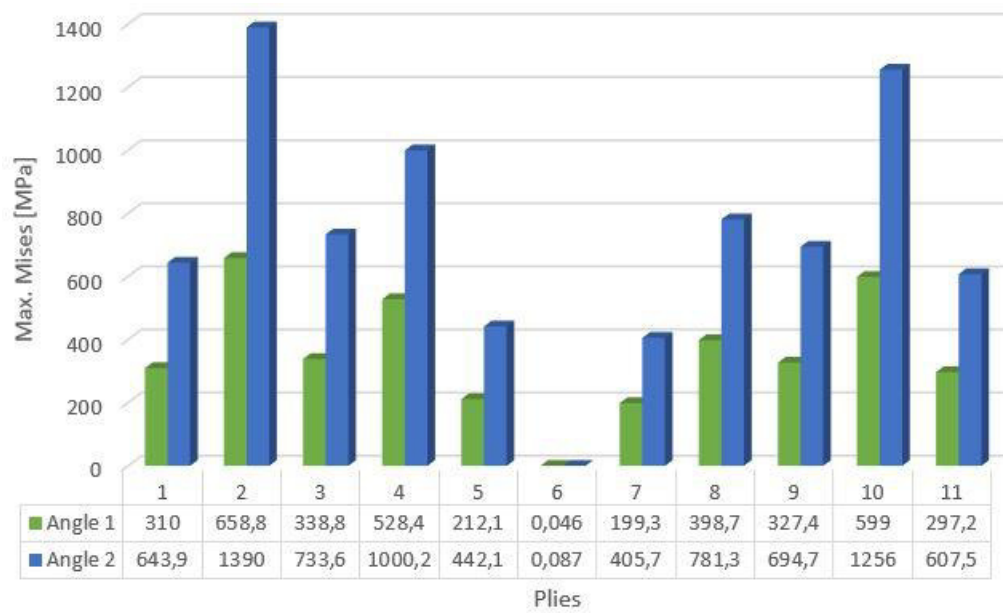


Fig. 6 Graph of results

## 5. Conclusion

When comparing two computing, geometrically identical models we found out that different stress in the individual layers was shown at the different material properties orientation. As might be seen in the Table 2, the material layer orientation at composite materials apparently influences their behavior when under the pressure. A layer with the same material and the same thickness shows various values of stress due to the influence of the material properties orientation. This phenomenon may be used composites configuration for various types of pressure. By doing this we may prevent useless material re-dimensioning and

we may achieve maximum usage of the material properties. It is possible to optimally set the composition and arrangement of composite material by setting the correct orientation of material properties.

## References

- [1] Abaqus 6.13 documentation. <http://129.97.46.200:2080/v6.13/>
- [2] Jancár J. Úvod do materiálového inžinýrství polymerních kompozitu (Introduction to materials engineering of polymer composites). VUT in Brno. Chemical faculty; 2003
- [3] Jones RM. Mechanics of composite material. ISBN 1-56032-712-X; 1998
- [4] Kořínek Z. Kompozity. <http://www.volny.cz/zkorinek/>
- [5] Matthews FL. Rawlings RD. Composite materials: engineering and science. Chapman and Hall. London; 1994
- [6] Kratochvíl B. Švorčík D. Úvod do studia materiálů (Introduction to the study of materials) VŠCHT. Prague; 2005
- [7] Kochaník J. Pavlenko S. Uram V. Haľko J. Teória pružnosti a plasticity. FVT TU v Košiciach so sídlom v Prešove; ISBN 80 – 7099 – 977 – 2; 2003
- [8] Kompiš V. Murčínková Z. Droppa P. Computational simulations of toughening mechanisms for composite materials of short and middle – large – aspect – ratioreinforced composites. In composite materials in theory and practice. VŠB – TU Ostrava; 2013
- [9] Laš V. Mechanika kompozitních materiálů (Mechanics of composite materials). University of West Bohemia, Pilsen; 2008
- [10] Pavlenko S. Murčínková Z. Uram V. Kochaník J. Metóda konečných prvkov. Teória a príklady. FVT TU v Košiciach so sídlom v Prešove; ISBN 80 – 8073 – 246 – 9; 2004
- [11] Tenek LT. Argyris J. Finite Element Analysis for Composite Structures. Kluwer Academic Publishers; 1998
- [12] Vrbka J. Mechanika kompozitu. FSI VUT in Brno; 2008
- [13] Žmindák M. Grajciar I. Nozdrovický J. Modelovanie a výpočty v metóde konečných prvkov. SĽF UNIZA in Žilina; 2004